

**The Study of the Effect of Impeller Shape and Location on the Mixing Behaviour
Using CFD Modeling**

by

Mohd Hafizudin Bin Rozali

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Chemical Engineering)

JULY 2010

Universiti Teknologi PETRONAS,
Bandar Seri Iskandar,
31750 Tronoh ,
Perak Darul Ridzuan.

CERTIFICATION OF APPROVAL

**The Study of the Effect of Impeller Shape and Location on the Mixing Behaviour
Using CFD Modeling**

by

Mohd Hafizudin Bin Rozali

A project dissertation submitted to the
Chemical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(CHEMICAL ENGINEERING)

Approved by,

(Dr. Ku Zilati Ku Shaari)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

July 2010

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

MOHD HAFIZUDIN BIN ROZALI

TABLE OF CONTENTS	PAGE
CERTIFICATION OF APPROVAL.	.ii
CERTIFICATION OF ORIGINALITY.	.iii
ABSTRACT.	.viii
ACKNOWLEDGEMENT.	.ix
CHAPTER 1: INTRODUCTION.	.1
1.1 Project Background.	.1
1.2 Problem Statement.	.2
1.3 Objectives.	.3
1.4 Scope of study.	.3
CHAPTER 2: LITERATURE REVIEW.	.4
2.1 Mixing (process engineering).	.4
2.2 Impeller.	.4
2.3 Computational Fluid Dynamics (CFD).	.7
2.3.1 Applications of CFD.	.8
2.3.2 Advantages of CFD.	.8
2.3.3 Limitations of CFD.	.9
2.4 Grid Design (Cited from www.chmltech.com/cfd/grid_generation.pdf).	.10
2.5 Time Step Size.	.11
CHAPTER 3: METHODOLOGY.	.12
3.1 Methodology/Project Work.	.12
3.2 Preliminary Simulation.	.13
3.2.1 Problem Description.	.13
3.3 Settling Time.	.16
3.4 Meshing the Impeller.	.21

CHAPTER 4:	RESULT AND DISCUSSION.24
4.1	Different Shape of Impeller.24
4.1.1	Impeller1.24
4.1.2	Impeller2.26
4.1.3	Impeller3.28
4.1.4	Impeller4.30
4.2	Different Location (Distance from the Bottom of Tank) of Impeller.32
4.2.1	Lower Impeller.32
4.2.2	Middle Impeller.34
4.2.3	Upper Impeller.35
CHAPTER 5:	CONCLUSION AND RECOMMENDATION.37
REFERENCES.39
APPENDICES.41
LIST OF FIGURES..vi
LIST OF TABLES.vii

LIST OF FIGURES	PAGE
Figure 2.3.3: Comparison between poor and better velocity profile	9
Figure 3.1(a): Process flow of methodology	14
Figure 3.2(a): Problem specification	15
Figure 3.2(b): Grid/Mesh display of the mixing tank	15
Figure 3.3(a): Scale of range of volume fraction	16
Figure 3.3(b): Settling time	17
Figure 3.3(c): Time step size 0.001s, from 0s-220s	18
Figure 3.3(d): Time step size 0.002s, from 0s-230s	18
Figure 3.3(e): Time step size 0.005s, from 0s-230s	18
Figure 3.3(f): Time step size 0.01s, from 0s-280s	18
Figure 3.3(g): Time step size 0.02s, from 0s-340s	19
Figure 3.3(h): Time step size 0.05s, from 0s-450s	19
Figure 3.3(i): Time step size 0.1s, from 0s-1000s	19
Figure 3.3(j): Time step size 0.5s, from 0s-3500s	19
Figure 3.3(k): Graph time to reach settling time (s) vs. time step size (s)	20
Figure 4.1.1(a): Initial contour of sand volume fraction before simulated	24
Figure 4.1.1(b): Contours of volume fraction of sand from 1.0s to settling time of 23.0s	25
Figure 4.1.1(c): Velocity vector of sand at 23.0s	26
Figure 4.1.2(a): Initial contour of sand volume fraction before simulated	26
Figure 4.1.2(b): Contours of volume fraction of sand from 1.0s to settling time of 7.0s.	27
Figure 4.1.2(c): Velocity vector of sand at 7.0s	27
Figure 4.1.3(a): Initial contour of sand volume fraction before simulated	28
Figure 4.1.3(b): Contours of volume fraction of sand from 1.0s to settling time of 17.0s	28
Figure 4.1.3(c): Velocity vector of sand at 17.0s	29
Figure 4.1.4(a): Initial contour of sand volume fraction before simulated	30

Figure 4.1.4(b): Contours of volume fraction of sand from 1.0s to settling time of 15.0s	30
Figure 4.1.4(c): Velocity vector of sand at 15.0s	31
Figure 4.2.1(a): Contours of volume fraction of sand from 1.0s to settling time of 250.0s	32
Figure 4.2.1(b): Velocity vector of sand at 250.0s	33
Figure 4.2.2(a): Contours of volume fraction of sand from 1.0s to settling time of 210.0s	34
Figure 4.2.2(b): Velocity vector of sand at 210.0s	35
Figure 4.2.3(a): Contours of volume fraction of sand from 1.0s to settling time of 460.0s	35
Figure 4.2.3(b): Velocity vector of sand at 460.0s	36

LIST OF TABLES	PAGE
Table 3.4: Mesh file of all samples	21

ABSTRACT

This project presents on predicting the flow and mixing behaviour inside the mixing tank and to investigate the effect of impeller design or shape and location on the mixing characteristic. Impeller location is defined as the distance between the base of the tank and the surface below the impeller's blade. It is generally believed that mixing in stirred tank is dominated by the turbulent diffusion or dispersion compared to the convective mixing. Replace a new impeller inside the tank without properly tested, designed and modelling will cost much. If bad mixing occurs, it may lead to increase cost due to further modification and the production also will decrease. Prototype also incurred cost to build and operate it and require looking like the same as the real one. Therefore, in this project, FLUENT 12.0.6 will be used to simulate the flow inside the mixing tank. The impeller shape and location will be investigated to know how they will affect the flow and the mixing behaviour inside the mixing tank. The existing mesh created by GAMBIT® for the tutorial will be re-used and modified its impeller design and to simulate the flow by using FLUENT®. The objective of this project is to predict the result which design for the impeller is the best for better and efficient mixing in the tank. It is clearly shown that the lower impeller location inside the tank gives better mixing condition than others. The time to reach settling time for this impeller's location also is optimum and acceptable. Recreate the impeller shape cause the weight of impeller to increase thus reducing the impeller's rotational speed. This will lessen the efficiency of the mixing tank.

ACKNOWLEDGEMENT

The first and foremost, I would like to convey my gratitude to God for giving me enough time, courage, strength and determination in completing this research. Without the help from Him, I will not be able to perform outstandingly. To my dearest parent and family members who are always there to motivate me in achieving my dream and goals in education, your motivation and never ending love is the thing that makes me stand today.

To my supervisor, Dr Ku Zilati Ku Shaari who has assisted me a lot in finishing the research. The discussions that I had with you have given me fruitful information in pursuing my research works. Your generous heart in giving me the permission to use some of your research lab materials is highly appreciated. Without that, it will hard for me to continue my work.

Besides, gazillion thanks to all the lecturers who taught me a lot during my study here. The knowledge passed is priceless. To all my fellow friends in UTP, these people have inspired to strive for my level best in order to finish my education here successfully. I have accomplished my dreams with the kind and generous help from all of you. Thank you.

CHAPTER 1

INTRODUCTION

1.1 Project Background

Mixing tank is widely used in chemical, pharmaceutical, food and metallurgical process industries as well in municipal and industrial wastewater treatment. Mixing has important part in various sectors of industry to obtain products of high added value. At an industrial scale, efficient mixing can be difficult to achieve. A great deal of engineering effort goes into designing and improving mixing processes. Mixing at industrial scale is done in batches or with help of static mixers.

Computational fluid dynamics (CFD) is playing a key role in helping to understand the flow inside the mixing tank. The CFD has been used in the last two decades to devise solutions and gain insight of the flow inside the mixing tank and CFD, together with experimental validation, has been able to improve the design of many reactor systems (Pedrosa and Nunhez, 2000).

In this work, a simulation of granular multiphase flow in a mixing tank is to be performed using Fluent®. The Eulerian multi-fluid model will be used along with the standard k- turbulence model to simulate solid-liquid flow in a tank.

The focus of this project is to predict the mixing behavior inside the mixing tank and to investigate the effect of impeller shape and location on the mixing characteristic. The location is defined as the distance from the base or bottom of the tank to the below surface of the impeller's blades. The problem domain is in two dimensional (2D) and axisymmetric which the mesh file is created or modified half of the mixing tank.

1.2 Problem Statement

It is generally believed that mixing in stirred tank is dominated by the turbulent diffusion or dispersion compared to the convective mixing. There were several factors that influenced the flow and hence the characteristics of mixing in the mixing tank. Various researches had been made on investigation or studying the behavior of mixing includes experimentation, theoretical studies and numerical methods by using CFD.

Modification or replace a new impeller inside the tank or to try several design of the impeller will cost much because the operation of the process will stop temporarily. Let say after that bad mixing occurs, it may lead to the increase of cost due to further modification and the production also will decrease due to operation has been stopped. Thus, this will result in less profit or no profit at all. Another aspect that should be observed is that industries today have to comply with safety and environmental regulations. Prototype also incurred cost to build and operate it and the design has to be as similar as possible to the real one.

By using CFD model, one can produce many design of tank without using money thus can achieve the best design to be applied in the real situation. In this project, FLUENT 12.0.16 will be used to simulate the flow inside the mixing tank. The impeller design/shape and location will be studied to investigate their effect on the flow and the mixing behavior inside the mixing tank. The existing mesh created by FLUENT® for the tutorial will be re-used and modified by using GAMBIT® to simulate the flow. The template of the grid for the mixing tank and the C-programme for the User Define Function is obtained from the tutorial also will be used as the basis for this simulation. CFD approach also will provide a virtual lab for company so that they will be able to conduct more research and development for their process and products.

1.3 Objectives

The objectives of this project are:

- To simulate the flow and mixing behavior based on the properties of the impeller:
 - Shape/design
 - Location (the distance of the impeller from the bottom tank)
- To predict the best impeller design for better and efficient mixing in the tank.
- To study time taken to reach perfect mixing & settling time inside the mixing tank.

1.4 Scope of study

Basically, the scope of study in this project is emphasizing more on the effect of the impeller properties on the mixing characteristic inside the mixing tank. Mixing tank is used to maintain solid particles or droplets of heavy fluids in suspension. Mixing may be required to enhance reaction during chemical processing or to prevent sedimentation. The properties of the impellers such as impeller shape and the location of the impeller will be investigated in this project. The relationship between settling time and perfect mixing also will be investigated in this project.

CHAPTER 2

LITERATURE REVIEW

2.1 Mixing (process engineering)

In industrial process engineering, mixing is a unit operation that and its purpose is to make a heterogeneous mixture or system turn into a homogeneous mixture. Familiar examples include pumping of the water in a swimming pool to homogenize the water temperature, and the stirring of pancake batter to eliminate lumps. In this recent year, many researches had been made on improving the design of many reactor systems.

One example of a mixing process in the industry is concrete mixing, where cement, sand, small stones or gravel and water is commingled to a homogeneous self-hardening mass, used in the construction industry. Another example is mulling foundry molding sand, where sand, bentonite clay, fine coal dust and water are mixed to a plastic, moldable and reusable mass, applied for molding and pouring molten metal to obtain sand castings that are metallic parts for automobile, machine building, construction or other industries (Wikipedia, 2009).

2.2 Impeller

Impellers in agitated tanks are used to mix fluids or slurry in the tank. This can be used to combine materials between solids, liquids, and gas. Mixing the fluids in a tank is very important if there are gradients in conditions such as temperature or concentration.

There are two types of impellers which depend on the flow regime created which are axial flow impeller and radial flow impeller. Radial flow impellers impose shear stress to the fluid, and are used, for example, when we need to mix immiscible liquids. Another application of radial flow impellers are the mixing of very viscous fluids. Axial flow impellers impose essentially bulk motion, and are used on homogenization processes, in which is important to increase fluid volumetric flow rate.

One component inside the mixing tank that is affect the flow and mixing behavior is the impeller. Several efforts have been made in the past to improve the performance of different impellers in terms of their power consumption, pumping capacity and degree of mixing achieved and new types of impellers are continuously being evolved (see, for example, Li et al., 2004; Mavros et al., 2001). Mixing in a stirred vessel is achieved by three means: at the molecular level, due to turbulence fluctuations and due to bulk convection or circulation of the fluid. Various parameters, such as the impeller shape, impeller diameter, number of impeller blades, tank diameter, clearance of the impeller from the tank bottom are known to affect significantly the flow within the vessel and hence the mixing characteristics (Buwa et al., 2006).

In stirred vessel, the quality of flow generated by the impeller mainly depends upon the impeller design. The ongoing demand for the improved impeller designs usually comes from the users of industrial mixing equipment when the vessels are to be design for new plants or improvement in the existing design is desired for enhancing quality, capacity, process efficiency, and energy efficiency (Kumaresan and Joshi, 2006).

The effect of impeller designs and its speed has been investigated in depth by a few research groups. For example, the article by Kumaresan and Joshi (2006) has made unique contributions in this area. In this article, a combination of LDA measurements and CFD predictions using sliding mesh approach has been performed to study the effect of impeller design and mixing time for a set of axial flow impellers: pitched blade turbines and hydrofoils. In conclusion, a very good agreement has been observed between experimental and the predicted mixing time over a wide range of impellers varying in number of blades, blade angle, blade width and impeller diameter (Kumaresan and Joshi, 2006).

The other article worked on anchor type impellers. CFD helped to see that anchor impellers are particularly suited for pseudo plastic fluids because it promotes mixing at the close clearance between blades and wall, where flow is generated. It shows that mixing can be improved by the increase of rotation speed and by the use of impellers with a higher blade height and with a blade support (Pedrosa and Nunhez, 2000). The

article on fluid dynamics and mixing in a stirred vessel with a grid disc impeller is conducted both experimental and numerical investigations. The three dimensional rotational turbulent flow was computed using the Reynolds-averaged Navier-Stokes equations. Measurements of profiles of mean velocity components, turbulent kinetic energy, mixing time and power consumption were performed and compared with the predictions. The performance of the proposed grid disc impeller is comparable to that of a standard impeller which is known to have low power requirement (Buwa et al., 2006).

Research on Ekato Intermig impellers was also done to analyze flow and mixing in a vessel. Three impeller speeds are considered in the experiment. It is showed that despite the complex flow field created by the impellers, homogeneous mixing is not achieve even after several hours of processing time if highly viscous materials are used (E.S. Szalai et al., 2004). It is evident that the impeller speed required for suspension by a pitch blade turbine downflow (PBTd) impeller is much lower than required by pitch blade turbine upflow (PBTu) and disc turbine (DT) impellers. Further, for DT and PBTu impellers the present CFD model predicts a significant quantity of unsuspended particles present on the tank bottom. It can be noted that with an increase in the impeller rotational speed the amount of solid particles present at the bottom of the reactors has decreased (Murthy et al., 2007).

In this project, the impeller shape and location will be studied to investigate their effect on the flow and the mixing behavior in the mixing tank. All the shapes or types of the impeller from the articles above will be used thus comparison can be made in order to achieve the best design of the impeller. FLUENT® will be used to simulate the multiphase flow in a mixing tank and the design of the impeller will be created by using GAMBIT®.

2.3 Computational Fluid Dynamics (CFD)

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics studies. It involves numerical methods and algorithms to solve the problems related to fluid flows. Basically, computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions (Wikipedia, 2008).

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The result of CFD analyses is relevant engineering data used in (André Bakker, 2002-2008):

- Conceptual studies of new designs.
- Detailed product development.
- Troubleshooting.
- Redesign.

The fundamental basis of almost all CFD problems is the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion.

Below are the general basic procedure used in all approaches in CDF:

- During preprocessing
 - The geometry (physical bounds) of the problem is defined.
 - The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non uniform.

- The physical modeling is defined , for example, the equations of motions + enthalpy + radiation + species conservation
 - Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.
- The simulation is started and the equations are solved iteratively as a steady-state or transient.
- Finally a postprocessor is used for the analysis and visualization of the resulting solution.

2.3.1 Applications of CFD

CFD is widely used especially in industrial processes to study the flow and heat transfer inside boilers, heat exchangers, combustion equipment, pumps, blowers, and piping. CFD also is used in aerodynamics of ground vehicles, aircraft, and missile. The other applications of CFD are (André Bakker, 2002-2008):

- Film coating, thermoforming in material processing applications.
- Flow and heat transfer in propulsion and power generation systems.
- Ventilation, heating, and cooling flows in buildings.
- Chemical vapor deposition (CVD) for integrated circuit manufacturing.
- Heat transfer for electronics packaging applications.

2.3.2 Advantages of CFD

Based on the lecture slides by André Bakker (2002-2008), there were a few advantages of using CFD. It is so obvious that using CFD is relatively low cost. This is because using physical experiments and tests to get essential engineering data for design can be expensive. CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful. In addition, CFD simulations can be executed in a short period of time and quick turnaround means engineering data can be introduced

early in the design process. CFD also has the ability to simulate real conditions due to many flow and heat transfer processes cannot be easily tested.

2.3.3 Limitations of CFD

André Bakker (2002-2008) also stated a few limitations of using CFD which are:

- Physical models.
 - CFD solutions rely upon physical models of real world processes such as turbulence, compressibility, chemistry, multiphase flow, etc.
 - The CFD solutions can only be as accurate as the physical models on which they are based.
- Numerical errors.
 - Solving equations on a computer invariably introduces numerical errors.
 - Round-off error due to finite word size available on the computer. Round-off errors will always exist even though they can be small in most cases.
 - Truncation error due to approximations in the numerical models. Truncation errors will go to zero as the grid is refined. Mesh refinement is one way to deal with truncation error.
- Boundary conditions.
 - As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.
 - Example: flow in a duct with sudden expansion. If flow is supplied to domain by a pipe, you should use a fully-developed profile for velocity rather than assume uniform conditions.

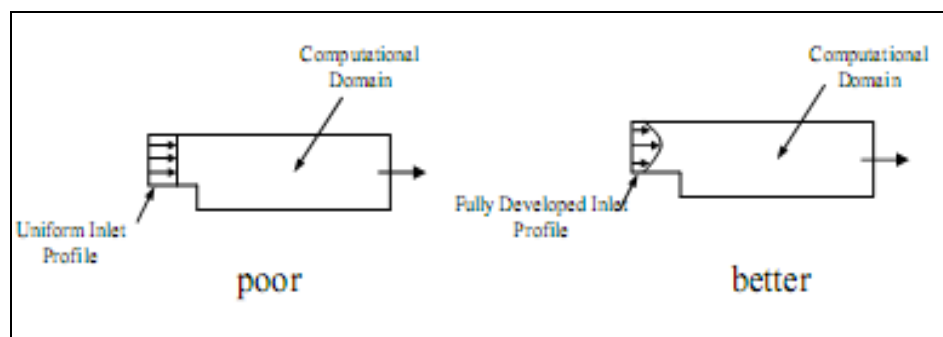


Figure 2.3.3: Comparison between poor and better velocity profile.

2.4 Grid Design (Cited from www.chmltech.com/cfd/grid_generation.pdf)

Grid generation is often considered as the most important and most time consuming part of CFD simulation. The quality of the grid plays a direct role on the quality of the analysis, regardless of the flow solver used. It is a great deal of importance to draw a top quality mesh to obtain a good solution. A few aspects such as the grid density, adjacent cell length or volume ratios, skewness, boundary layer mesh and mesh refinement through adaption should be considered to obtain a good quality mesh.

Basically there are three types of method in grid generation. It is important for the CFD analyst to know and understand all of the various grid generation methods. Only by knowing all the methods he or she can select the right tool to solve the problem at hand. There are Structured Grid Methods, Unstructured Grid methods and Hybrid Grid methods.

Structured grid methods take their name from the fact that the grid is laid out in a regular repeating pattern called a block. These types of grids utilize quadrilateral elements in 2D and hexahedral elements in 3D in a computationally rectangular array. Although the element topology is fixed, the grid can be shaped to be body fitted through stretching and twisting of the block. Unstructured grid methods utilize an arbitrary collection of elements to fill the domain. Because the arrangement of elements has no discernible pattern, the mesh is called unstructured. These types of grids typically utilize triangles in 2D and tetrahedral in 3D. While there are some codes which can generate unstructured quadrilateral elements in 2D, there are currently no production codes which can generate unstructured hexahedral elements in 3D.

Hybrid grid methods are designed to take advantage of the positive aspects of both structured and unstructured grids. Hybrid grids utilize some form of structured grid in local regions while using unstructured grid in the bulk of the domain. Hybrid grids can contain hexahedral, tetrahedral, prismatic, and pyramid elements in 3D and triangles and quadrilaterals in 2D. The various elements are used according to their strengths and

weaknesses. Hexahedral elements are excellent near solid boundaries where flow field gradients are high and afford the user a high degree of control, but is time consuming to generate. Prismatic elements usually triangles extruded into wedges are useful for resolving near wall gradients, but suffer from the fact that they are difficult to cluster in the lateral direction due to the underlying triangular structure.

The cell shapes for this project will be a 2D prism because to limit the calculation time of numerical method on the problem and to lessen the burden on the solver capabilities and simplify the problem. Take note also the sources of error which could prevent us from drawing a good mesh is when the mesh too coarse, have a high skewness, contain a large jumps in volume between adjacent cells, the mesh itself has a large aspect ratios, the interpolation errors at non-conformal interfaces and lastly the inappropriate boundary layer created for the mesh.

2.5 Time Step Size

The time step size is the magnitude of Δt . Since the FLUENT formulation is fully implicit, there is no stability criterion that needs to be met in determining Δt . However, to model transient phenomena properly, it is necessary to set Δt at least one order of magnitude smaller than the smallest time constant in the system being modeled. A good way to judge the choice of Δt is to observe the number of iterations FLUENT needs to converge at each time step. The ideal number of iterations per time step is 5-10. If FLUENT needs substantially more, the time step is too large. If FLUENT needs only a few iterations per time step, Δt should be increased. Frequently a time-dependent problem has a very fast "startup" transient that decays rapidly. Therefore, it is often wise to choose a conservatively small Δt for the first 5-10 time steps. Δt may then be gradually increased as the calculation proceeds (Fluent 6.3 Users Guide, 2006).

CHAPTER 3

METHODOLOGY

In this project, all simulation works will be performed by using FLUENT® software. Preliminary simulation will be conducted to simulate the granular multiphase flow in the mixing tank (FLUENT® Software Tutorial Guide, 2001). The design of the mixing tank will be in two dimensions (2D). The simulation will only be stopped if the volume fraction fulfils the requirement of settling time in the tank.

The simulation will be divided into two main parts. The first part will be simulation on the different impeller shapes. The second part is to simulate the flow with different impeller location. In this case, it is the distance between the below surface of the impeller's blade and the bottom of the tank. Three different locations will be investigated during the simulation. Basically, GAMBIT® will be used to modified and design the shape and the location of the impellers and this will be the toughest one in this project.

3.1 Methodology/Project Work

The methodology is divided into three main parts which are Problem identification and Pre-Processing, Solver Execution and Post processing (Fluent Software Training, 2001). For the first part, modeling goals need to be determined and in this project the objective is to simulate and study the mixing behavior in a mixing tank with changes in impeller properties like shape and height. Then the domain of the model also must be identified whether the model is in 3D or 2D and planar or axisymmetric problem. The domain of the model in this project is 2D axisymmetric mixing tank. The mesh file or grid design will be created by using design software which is GAMBIT®. The initial design is the symmetrical half of the mixing tank.

The next main part is Solver Execution which is run the simulation by using FLUENT®. All simulation is using only one numerical method which is Eulerian-Eulerian multiphase model. Eulerian-Eulerian approach is suitable for modeling dispersed multiphase system which has a significance volume fraction of the dispersed phase. This situation suits the study which will be conducted on the mixing reactor. After all setting has been setup, computing and monitoring the solution will take place in order to get the result. The last part is Post processing step which in this part the result that obtained will be examined so that any consideration on optimizing the model can be made to improve the results.

3.2 Preliminary Simulation

3.2.1 Problem Description

Based on FLUENT® Software Tutorial Guide, 2006:

The problem involves the transient startup of an impeller-driven mixing tank. The primary phase is water, while the secondary phase consists of sand particles with a 111 micron diameter. The sand is initially settled at the bottom of the tank, to a level just above the impeller. A schematic of the mixing tank and the initial sand position is shown in Figure 3. The domain is modeled as 2D axisymmetric. The fixed-values option will be used to simulate the impeller. Experimental data provided by FLUENT tutorial no. 20 entitled Using the Eulerian Multiphase Model for Granular Flow are used to represent the time-averaged velocity and turbulence values at the impeller location. This approach avoids the need to model the impeller itself. These experimental data are provided in a user-defined function.

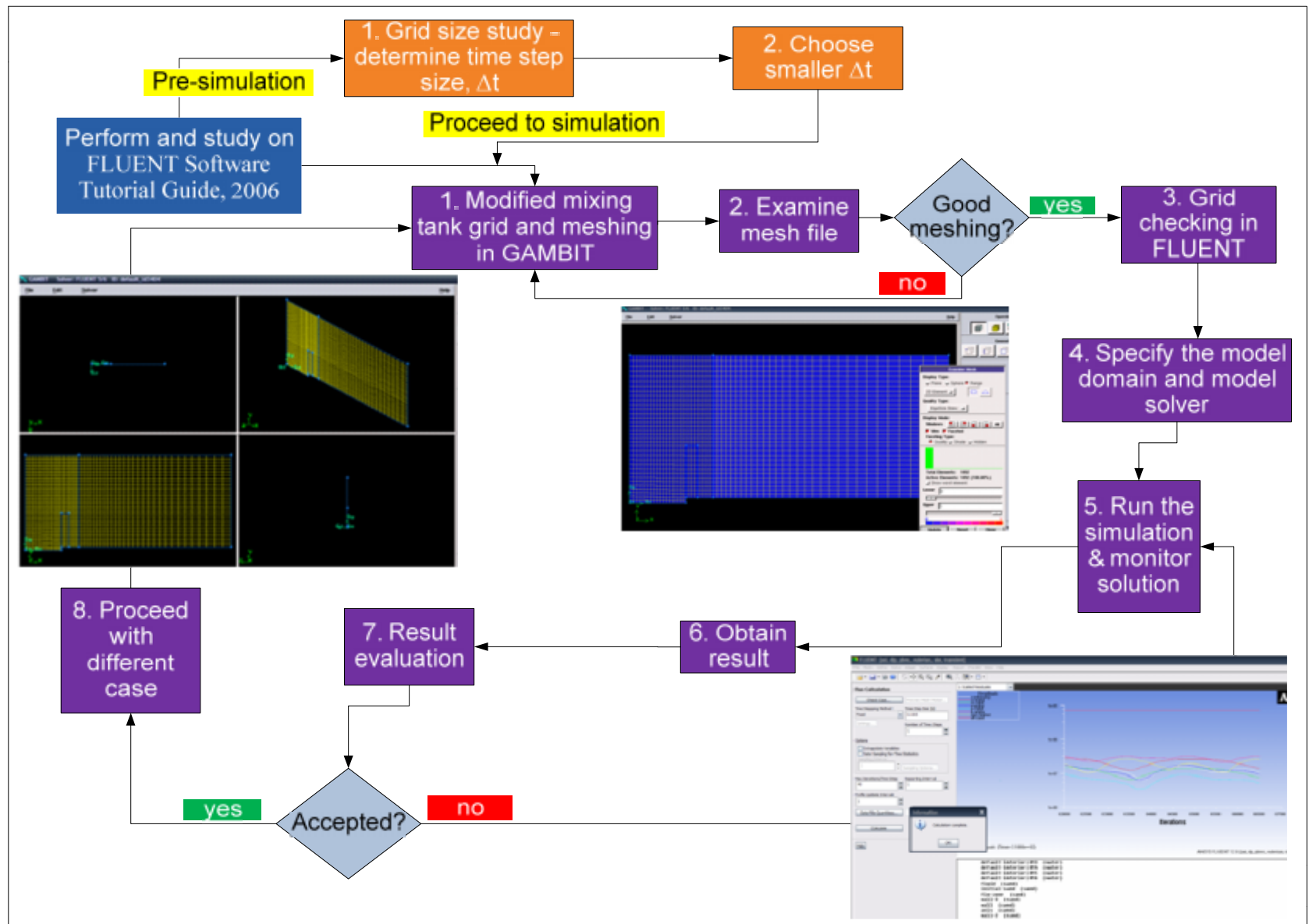


Figure 3.1(a): Process flow of methodology

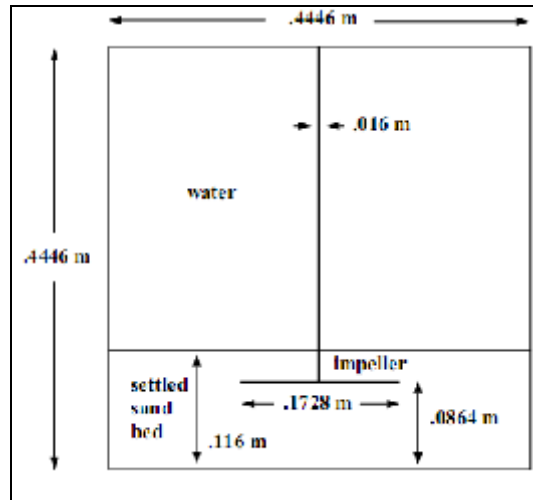


Figure 3.2(a): Problem specification

Based on the figure 3.2 (b) below, the existing mixing tank mesh file is taken from FLUENT tutorial. There are two phases inside the tank which are sand phase in black color and water phase in black color. The impeller is located in the middle of the tank. In GAMBIT, the mesh is created only in half shape and later in FLUENT the mirror plane will be used to get the full view of tank. The total active elements or cells in the mesh file are 1892. Since the mesh file is taken from tutorial, it is assumed to have a fine mesh or fine grid size.

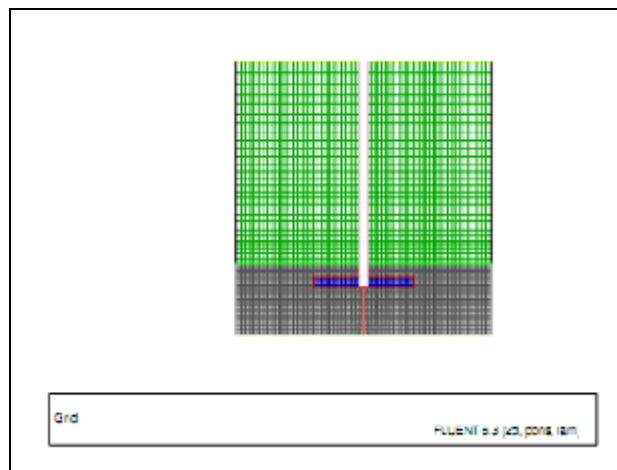


Figure 3.2(b): Grid/Mesh display of the mixing tank

3.3 Settling Time

The simulation will only be stop if the volume fraction fulfils the requirement of settling time in the tank. Settling time will be used in this study to indicate that the mixing is established in the mixing tank and does not exhibit any significant change in the mixing pattern after the settling time. Perfect mixing will be define as the state where the sand volume fraction is spread homogeneously in the mixing tank where the contour of sand consist of majorly (approximately more than 80%) of a single range of volume fraction from the scale shown below.

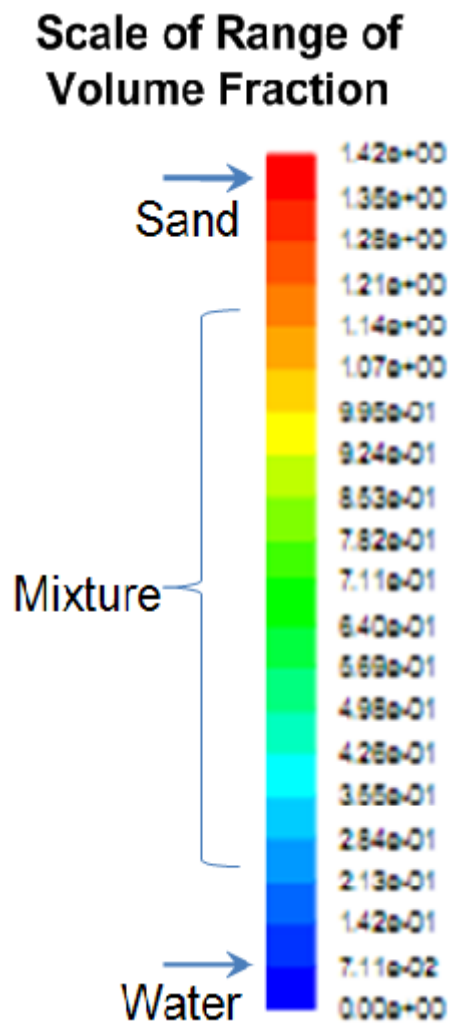


Figure 3.3(a): Scale of range of volume fraction

Settling time is the time required for an output to reach and maintain within a given error band following some input stimulus. The picture below showed the settling time of an amplifier or other output device is the time elapsed from the application of an ideal instantaneous step input to the time at which the amplifier output has entered and remained within a specified error band, usually symmetrical about the final value. In this project, settling time is the time for the mixing process to reach the phase or condition where there are no significant changes of the mixing pattern inside the tank.

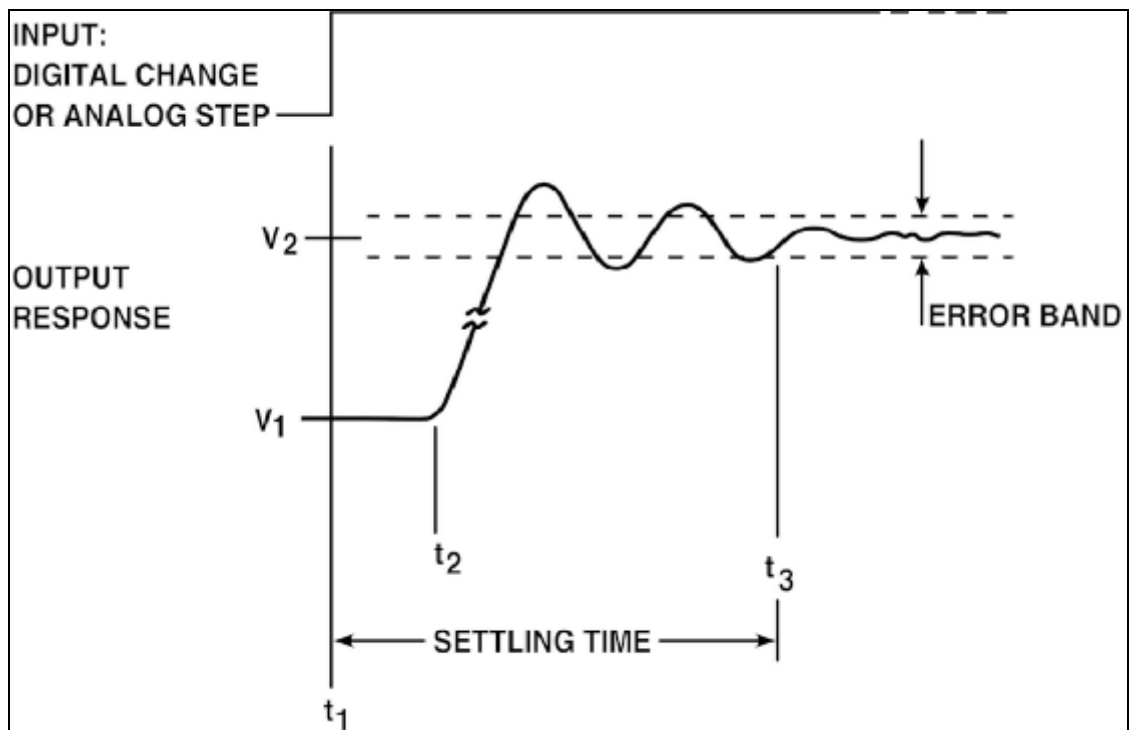


Figure 3.3(b): Settling time

In FLUENT, time step size determine how long the mixture in the tank will mix homogeneously or how long it will take to reach its settling time. In this project, there were six time step size that already applied while simulated the flow in the tank. The simulation started with the smallest size which is 0.001 s and then continued until a homogeneous mixing was obtained where the entire sand particle was evenly spread through water. After that, simulation of flow at 0.002 s, 0.005 s, 0.01 s, 0.02 s, 0.05 s, 0.1 s, and 0.5 s were conducted by the same procedure.

Below are the results (based on contour) of settling time for each time step size:

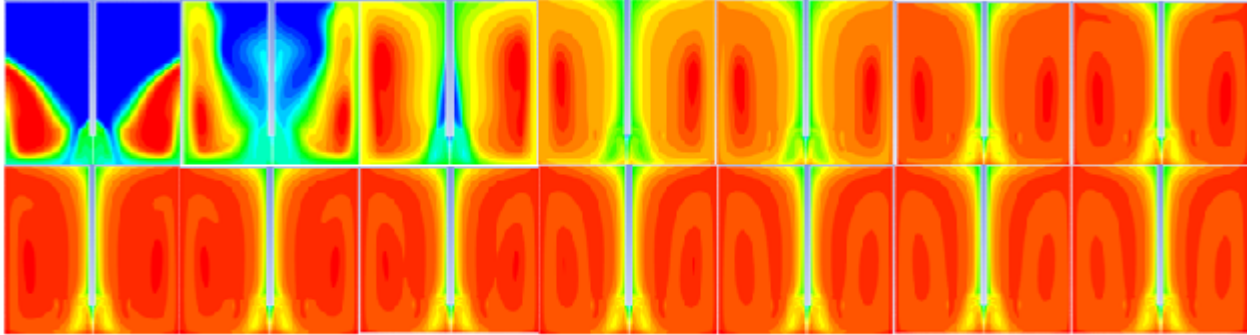


Figure 3.3(c): Time step size 0.001 s, from 0 s - 220 s

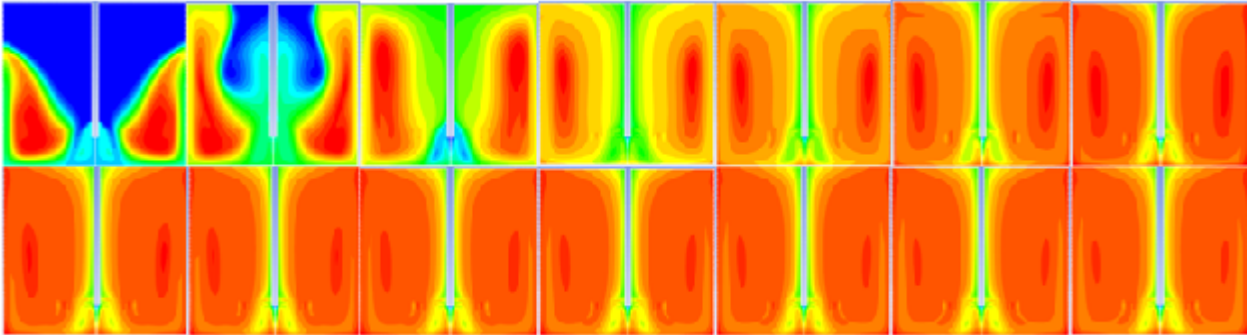


Figure 3.3(d): Time step size 0.002 s, from 0 s - 230 s

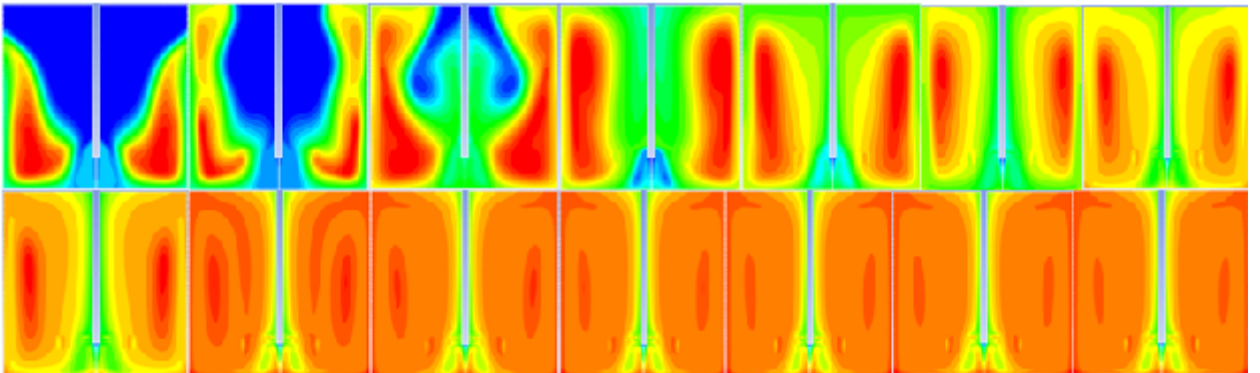


Figure 3.3(e): Time step size 0.005 s, from 0 s - 230 s

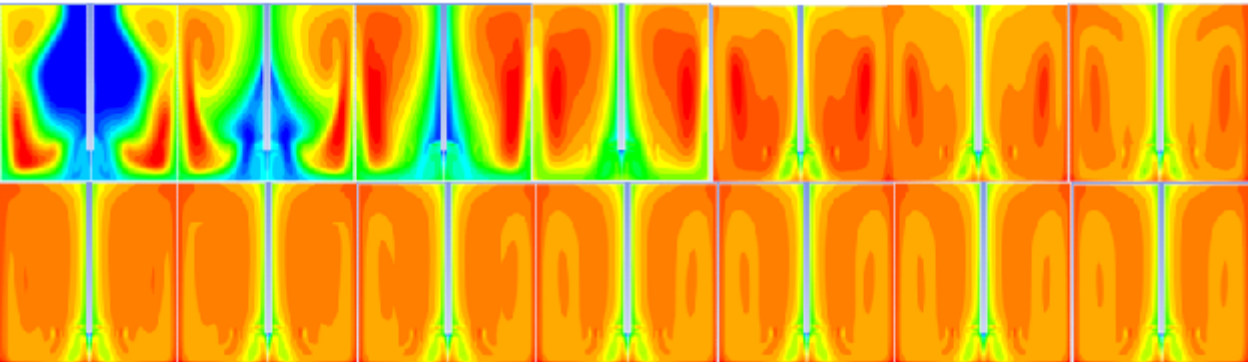


Figure 3.3(f): Time step size 0.01 s, from 0 s - 280 s

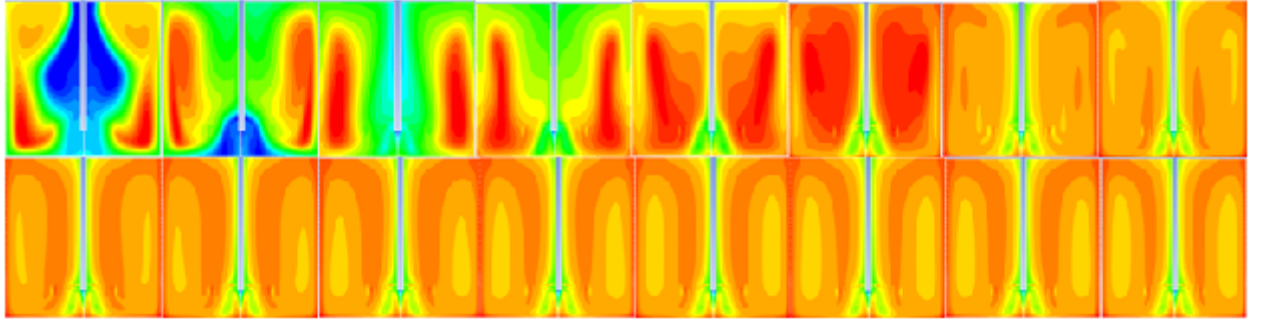


Figure 3.3(g): Time step size 0.02 s, from 0 s - 340 s

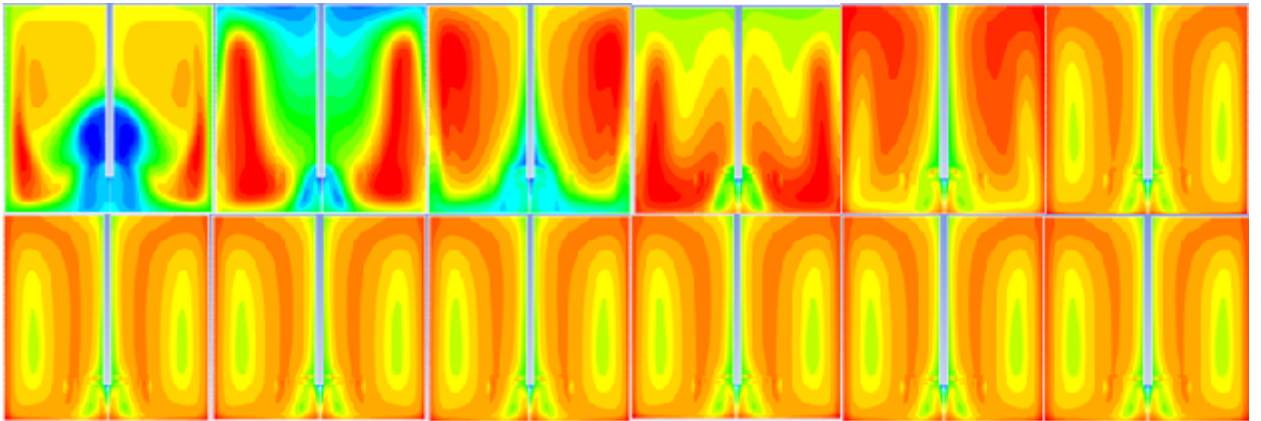


Figure 3.3(h): Time step size 0.05 s, from 0 s - 450 s

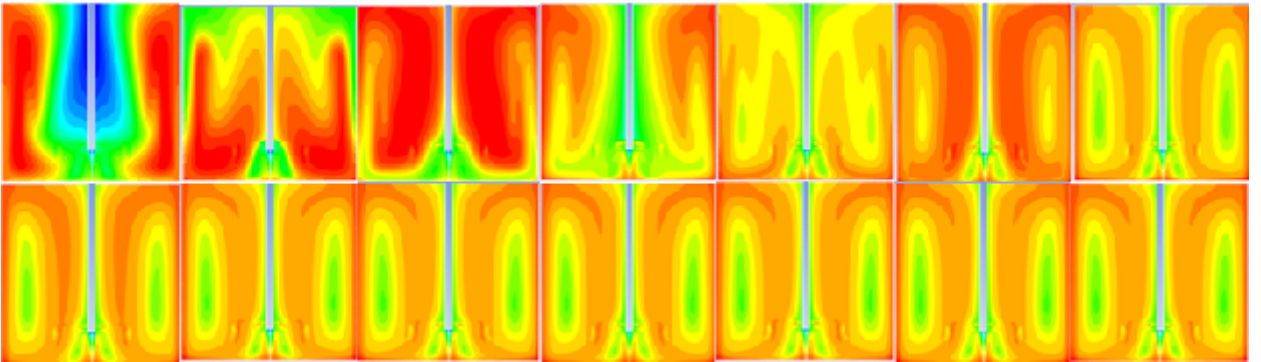


Figure 3.3(i): Time step size 0.1 s, from 0 s - 1000 s

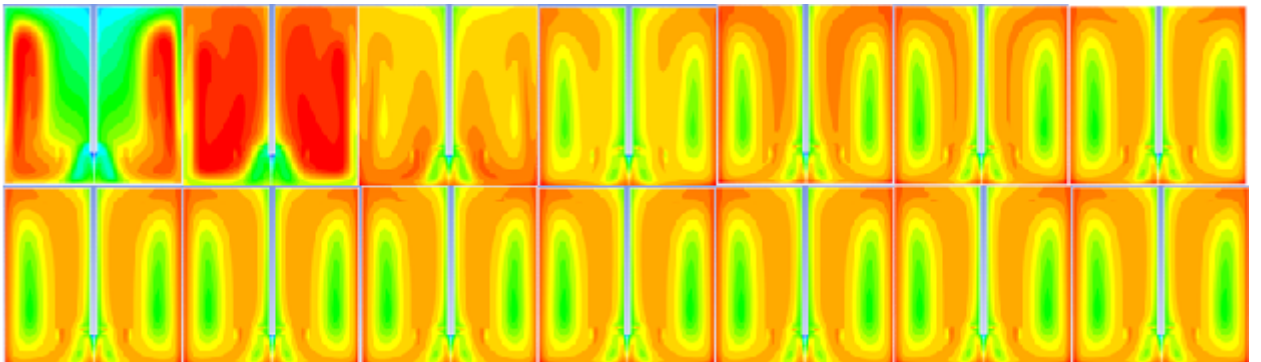


Figure 3.3(j): Time step size 0.5 s, from 0 s - 3500 s

All results were shown in the behavior of the volume fraction contour of the sand. The volume fraction settles at time when approximately more than 80% of the sand was equally dispersed in the water and there were no significant changes in the mixing pattern as being shown above for all time step size. Below is the graph time to reach settling time versus time step size. Based on the graph, the bigger time step size, the longer it will take to reach the settling time.

According to Rajesh and Yong (2009), for steady case, to validate the accuracy of the result, we have to check whether the mesh is refine enough. For unsteady case, we have another parameter that we have to take note of, which is the time step size. Since this project deals with unsteady state or transient model, the time step size should be considered in detail. The smaller the time step size, the more accurate the representation of the physical flow. Based on above description I have chosen 0.005 s as time step size in this project because time to reach settling time between 0.001 s, 0.002 s and 0.005 s time step size are about the same within the range of 220 s – 230 s. Since 0.005 s took shorter time to complete the simulation than 0.001 s and 0.002 s. In addition, it will give accurate results since the time step size is already quite smaller. Smaller time step means more accurate result but more computational time.

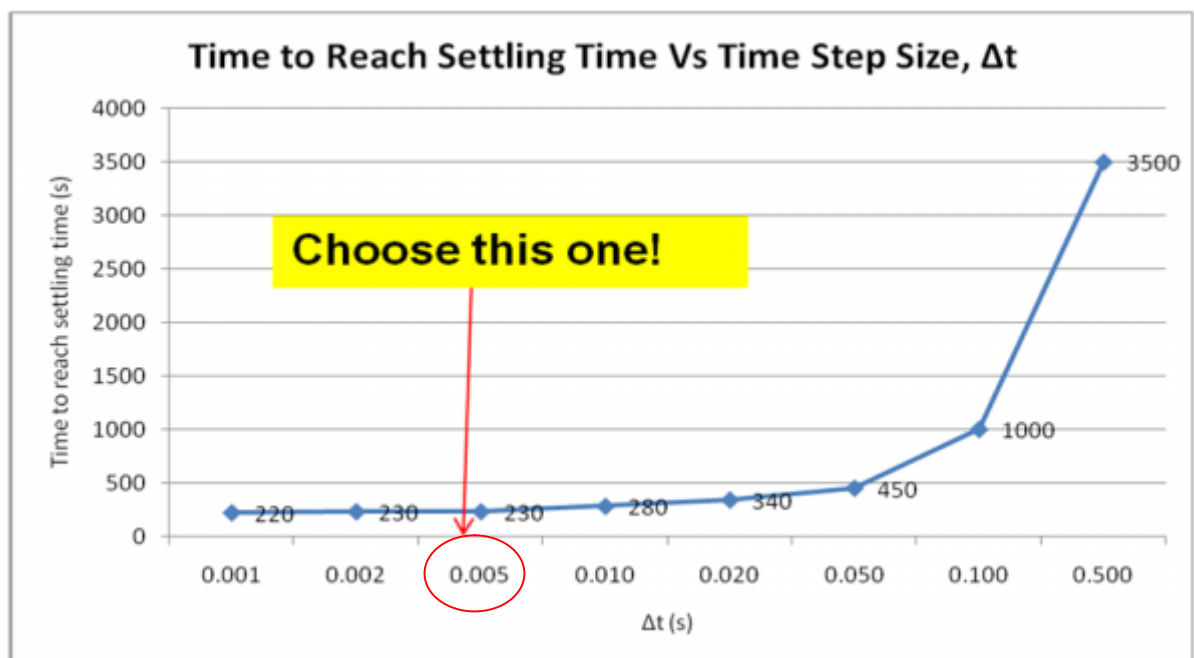
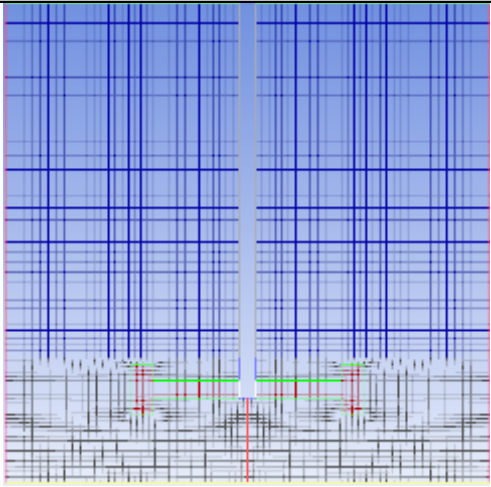
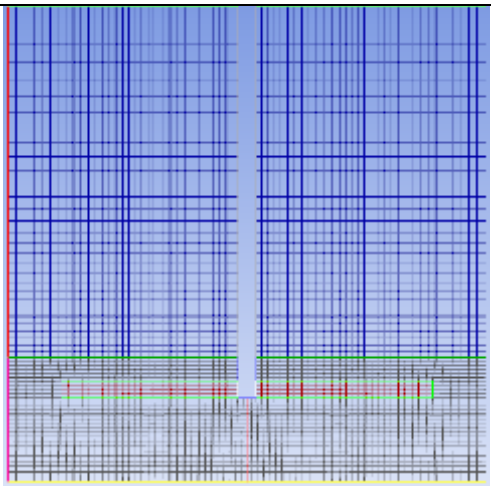


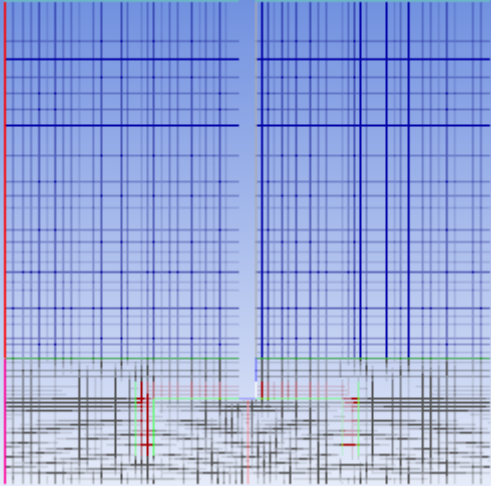
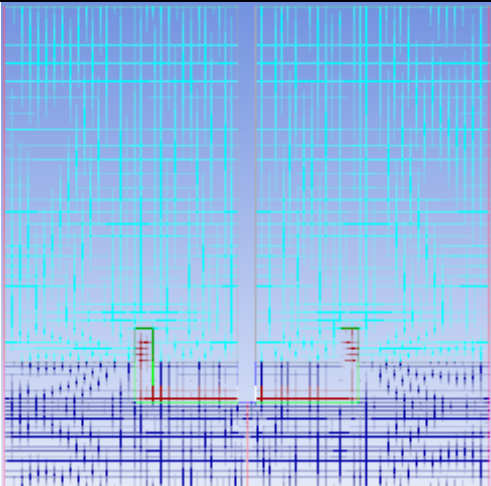
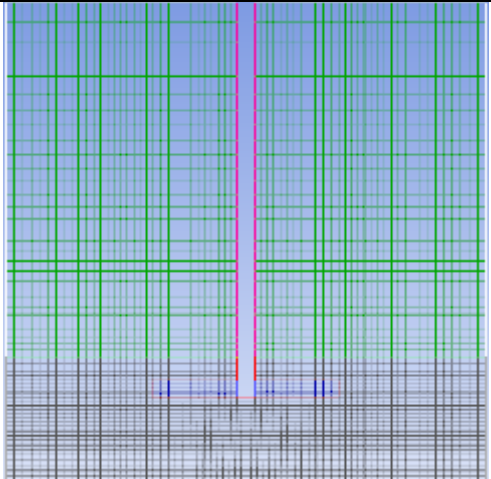
Figure 3.3(k): Graph time to reach settling time (s) vs. time step size (s)

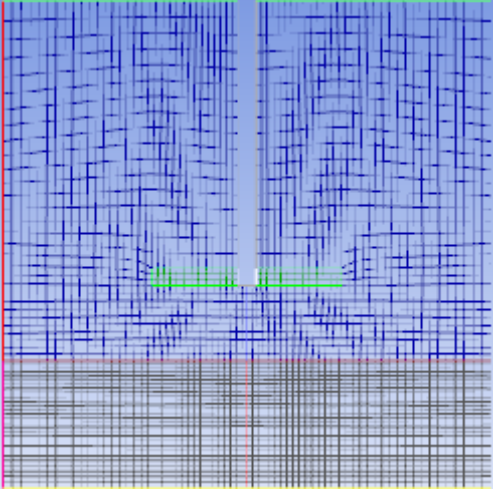
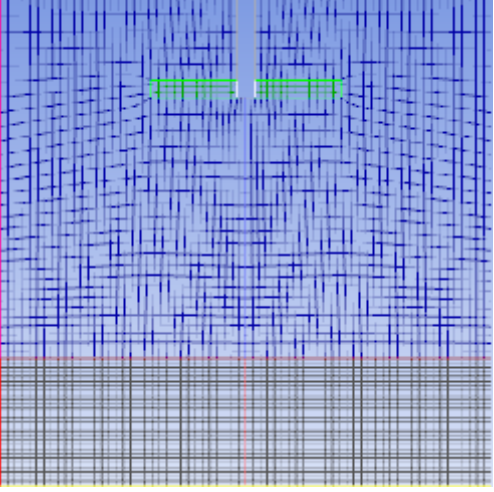
3.4 Meshing the Impeller

Based on the mesh file from the FLUENT tutorial 20, the impeller inside the tank will be modified by using GAMBIT. From all sample, it could be said that the shape of the impeller is modified to be longer in diameter and has been extended for different shapes. Since the mesh file is in 2D, it is difficult to create more complex and advance design for the impeller. Below are the picture of all shape and also different location of the impeller inside the mixing tank.

Table 3.4: Mesh file of all samples

Impellers' Shape	Mesh file
Impeller1 – Rushton-blade type	
Impeller2 – grid disc type	

<p>Impeller3 – anchor type (downward)</p>	
<p>Impeller4 – anchor type</p>	
<p>Impellers' Height</p>	<p>Mesh file</p>
<p>Lower position (0.0854m from the bottom of tank)</p>	

<p>Middle position (0.1847m from the bottom of tank)</p>	
<p>Upper position (0.3534m from the bottom of tank)</p>	

CHAPTER 4

RESULT AND DISCUSSION

4.1 Different Shape of Impeller

The first part of the simulation was to simulate the volume fraction of sand at different type of impeller. There are 4 types of impellers which are named as impeller1, impeller2, impeller3 and impeller4. The simulation will only be stopped if the volume fraction fulfils the requirement of settling time in the tank. Settling time will be used in this study to indicate that the mixing is established in the mixing tank. Perfect mixing will be defined as the state where the sand volume fraction spreads homogeneously in the mixing tank where the contour of sand consist of majorly (approximately more than 80%) of a single range of volume fraction from the scale shown before in **Figure 3.3(a)** and does not exhibit any significant change in the mixing pattern after the settling time. Settling time does not indicate perfect mixing but indicates mixing is in steady state where no changes in the mixing pattern.

4.1.1 Impeller1

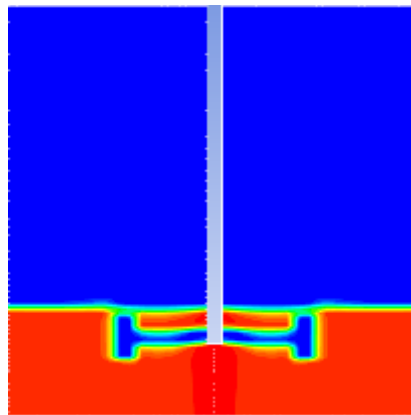


Figure 4.1.1(a): Initial contour of sand volume fraction before simulated

Below are the results (based on contour) of settling time for impeller1.

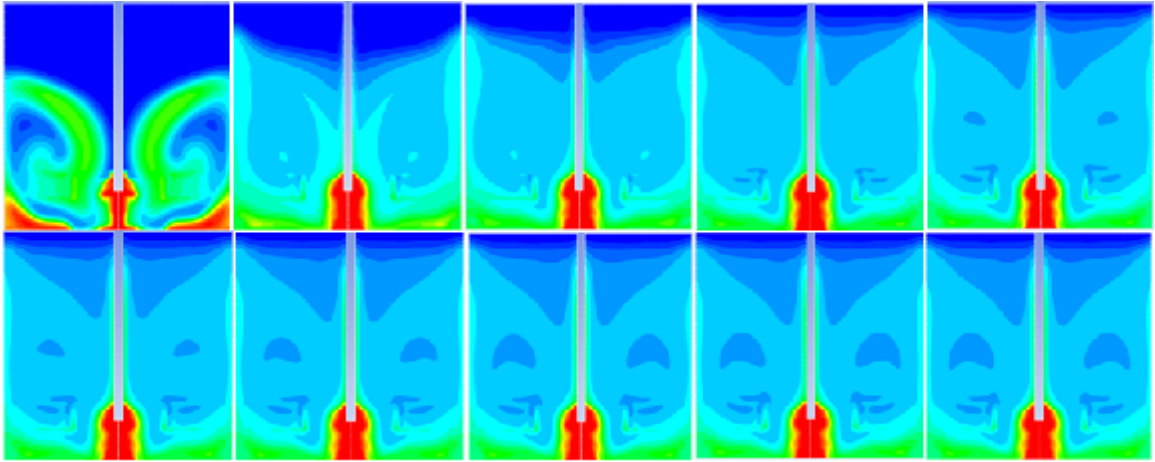


Figure 4.1.1(b): Contours of volume fraction of sand from 1.0s to settling time of 23.0 s

Based on the **Figure 4.1.1(b)**, the simulation was done at density of sand of 2500 kg/m^3 and with a Rushton-blade like impeller (front view). At the beginning of the simulation, the sand particles were spread up from the bottom to the side of the tank due to agitation and eventually rise to top to the maximum height of the tank upon reaching the settling time. It can be seen clearly that the turbulence pattern of the solid phase concentrates more on the side of the tank at the beginning of the process. After 23.0 s, there were no significant changes in the mixing pattern as it can be seen from the figure above and most of the sand was spread in the water phase. Thus, the process has reached its settling time which is at 23.0 s. Since the density of the sand particle is quite high which is 2500 kg/m^3 , the sand particles cannot reach up to the top of the tank and there were some amount trapped below the impeller and settled down at the bottom base of the tank. The last contour shows no perfect mixing achieved due to some amount of sand trapped below the impeller's blade. Notice that the action of the impeller draws clear fluid from above the originally settled bed and mixes it into the sand. To compensate, the sand bed is lifted up slightly. The maximum sand volume fraction has decreased as a result of the mixing of water and sand.

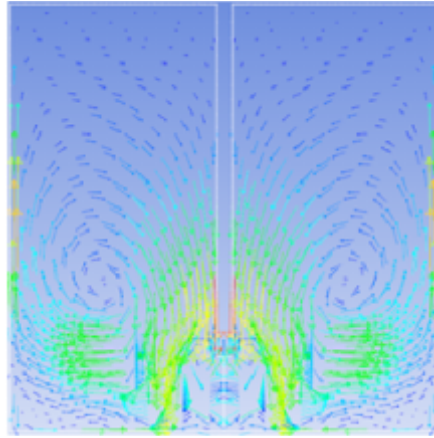


Figure 4.1.1(c): Velocity vector of sand at 23.0 s

Figure above shows velocity vector and magnitude of sand particles during the simulation process. It can be seen clearly that the region surrounding the impeller indicated circulation of sand in the water. The green color indicates that high magnitude of sand velocity move from the bottom of the tank to the side and then move upward before going down to the middle region of tank. The sand movements created a loop inside the tank. At top region of the tank, there is no presence of sand particle because the water velocity in that region is not sufficient to overcome the gravity force on the sand particles.

4.1.2 Impeller2

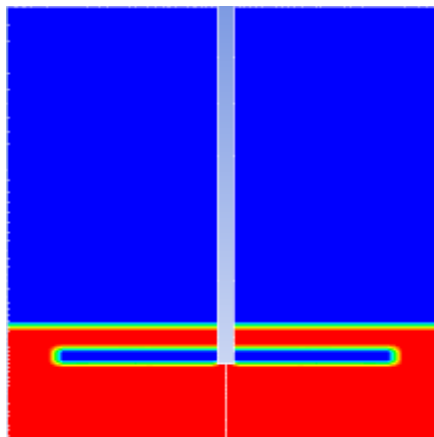


Figure 4.1.2(a): Initial contour of sand volume fraction before simulated

Below are the results (based on contour) of settling time for impeller2.

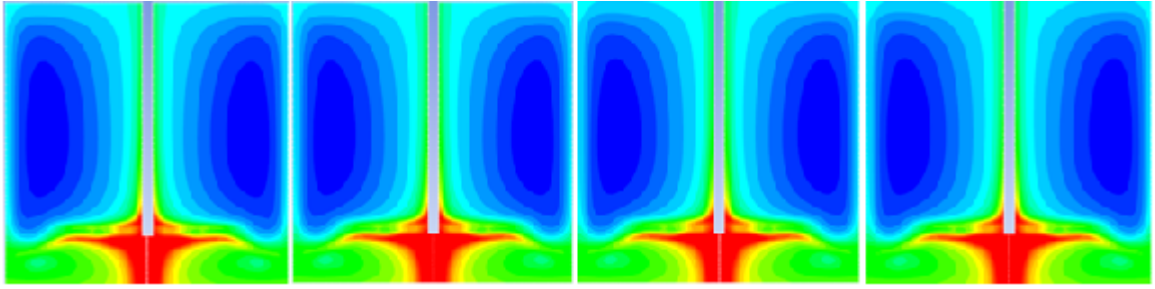


Figure 4.1.2(b): Contours of volume fraction of sand from 1.0 s to settling time of 7.0 s

Based on the **Figure 4.1.2(b)**, the simulation was done with the grid disc type impeller with longest diameter. It can be seen clearly that the flow pattern of the solid phase is restricted due to small opening at the side of the tank. The sand cannot move upward and only small amount of water mix with sand at the bottom of the tank. Homogeneous mixture cannot occur because of the flow restriction caused by the impeller. After 7.0 s, there were no significant changes in the mixing pattern as it can be seen from the figure above. Thus, the process has reached its settling time which is at 7.0 s. Due to flow restriction, the sand particles cant reaches even up to the middle of the tank and most of the sand particle trapped below the impeller and at the bottom base of the tank. No perfect mixing established for this type of impeller.

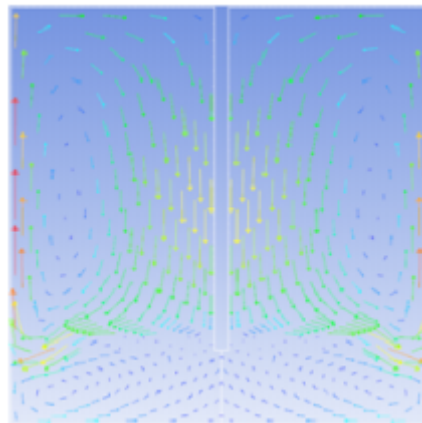


Figure 4.1.2(c): Velocity vector of sand at 7.0 s

From velocity vector and magnitude above, the highest velocity is at above region of the impeller which is mostly water phase. At region below the impeller, sand is still circulated and created a loop below the impeller. Only a small amount of sand at the side of tank can reached the middle region of the tank. The circulation of water is very strong in the middle region of the tank, though modest near the top.

4.1.3 Impeller3

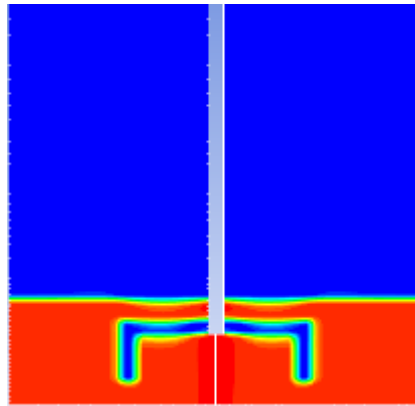


Figure 4.1.3(a): Initial contour of sand volume fraction before simulated

Below are the results (based on contour) of settling time for impeller3.

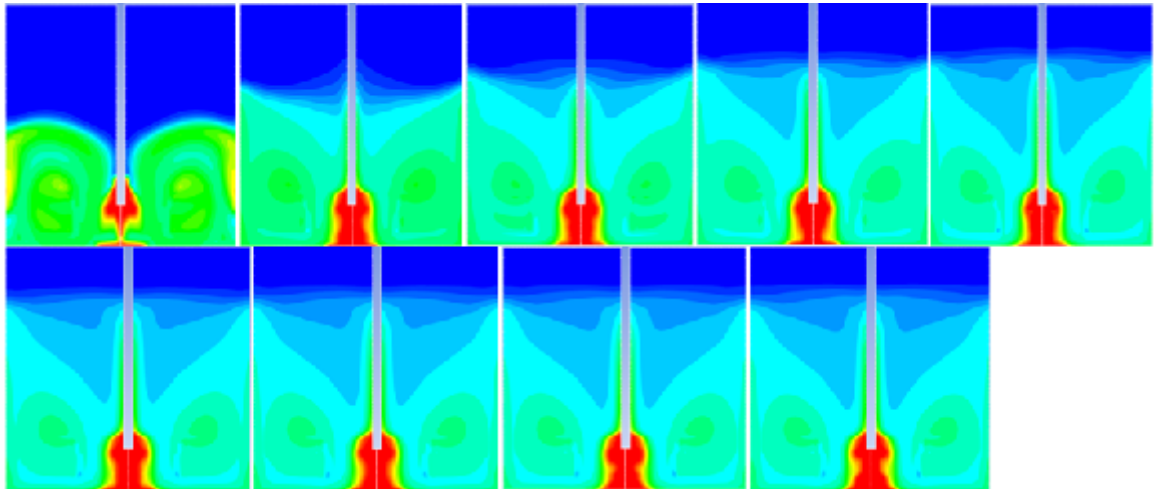


Figure 4.1.3(b): Contours of volume fraction of sand from 1.0 s to settling time of 17.0 s

The simulation was done with anchor type impeller but the side of the impeller is pointing downward. As in the picture above, at the beginning of the simulation the turbulence pattern is shown from the bottom up to the middle of the tank. As the simulation reached the settling time, the sand particles has been suspended much higher within the tank but not be able to reach up to the top of the tank due to high density of sand. The water velocity in the tank is not sufficient to overcome the gravity force on the sand particles. After 17.0 s, there were no significant changes in the mixing pattern and it can be said that the process already reached its settling time. Due to high density of sand, there was some amount of sand trapped below the impeller but it can be seen clearly that almost no sand particle settled at the bottom due to agitation resulted from the side of the impeller. Compare to the Rushton-blade type impeller, this type of impeller gave shorter time for the process to reach the settling time. No perfect mixing also established in the tank because the sand particles is not spread homogeneously in the water phase.

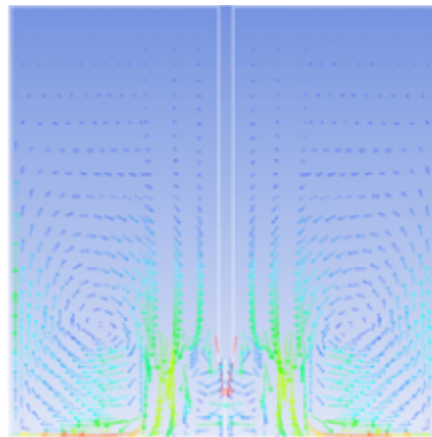


Figure 4.1.3(c): Velocity vector of sand at 17.0 s

Highest sand velocity also at the region near impeller and there is almost no activity at the top of the mixing tank due to high density of sand. At top region of the tank, there is no presence of sand particle because the water velocity in that region is not sufficient to overcome the gravity force on the sand particles.

4.1.4 Impeller4

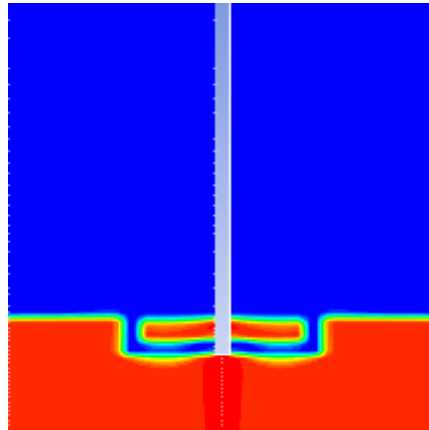


Figure 4.1.4(a): Initial contour of sand volume fraction before simulated

Below are the results (based on contour) of settling time for impeller4.

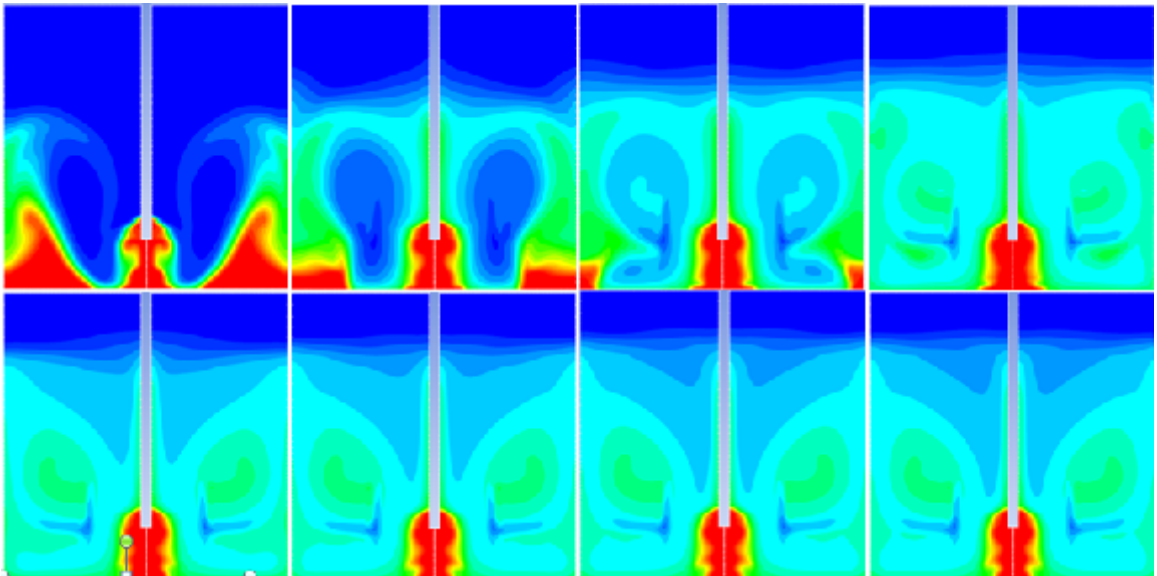


Figure 4.1.4(b): Contours of volume fraction of sand from 1.0 s to settling time of 15.0 s

The simulation was done with anchor type impeller but this time the side of the impeller is upward and the impeller location was in both phases. At the beginning of the simulation the turbulence pattern is shown from the bottom up to the side of the tank. After the sand particles reached the middle part then they went to the centre and went back to the impeller. There was a loop created around the side of the impeller. As the

simulation reached the settling time, the sand particles dispersed in the water but not be able to reach up to the top of the tank due to high density of sand. After 15.0 s, there were no significant changes in the mixing pattern and it can be said that the process already reached its settling time. Due to high density of sand, there was some amount of sand trapped below the impeller but it can be seen clearly that almost no sand particle settled at the bottom due to agitation resulted from the side of the impeller. The settling time for both anchor type impeller seem to be the same since there is only slightly different between both of them. In real industry this type of impeller has been used for highly viscous flow which is typical of polymer reactions and some processes in food industries. No perfect mixing also established in the tank because the sand particles is not spread homogeneously in the water phase.

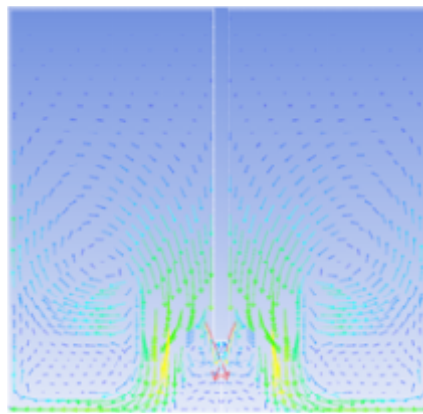


Figure 4.1.4(c): Velocity vector of sand at 15.0 s

Highest sand velocity also at the region near impeller and there is almost no activity at the top of the mixing tank due to high density of sand. At top region of the tank, there is no presence of sand particle because the water velocity in that region is not sufficient to overcome the gravity force on the sand particles.

4.2 Different Location (Distance from the Bottom of Tank) of Impeller

This is the second part of the simulation where impeller's location is being manipulated. The location is defined as the distance between the bottom or base of the tank and the impeller's blades. Value for each distance of the impeller is stated in the mesh file from **Table 3.4**. There were three locations of impeller, which is lower, middle and upper level inside the tank.

4.2.1 Lower Impeller

Below are the results (based on contour) of settling time for lower impeller.

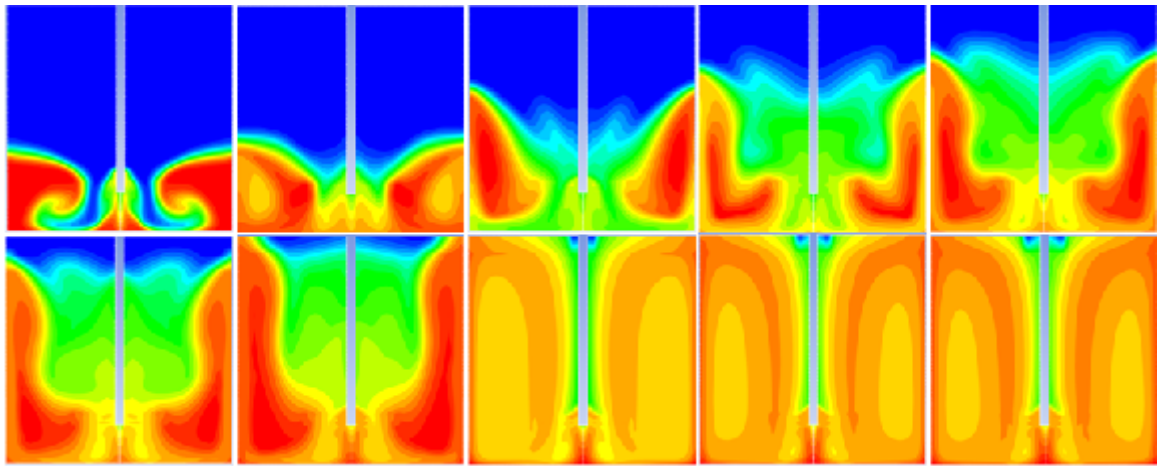


Figure 4.2.1(a): Contours of volume fraction of sand from 1.0 s to settling time of 250.0 s

The location of the impeller was within the sand phase. Based on the figure above, the sand particles were slowly dispersed into the water phase until homogeneous mixture form when the process reached its settling time. The solid particles circulated from the bottom to the side and up to the top of the tank. Then it will move downward toward the impeller. The original mesh file was used to simulate the flow inside the tank. After 250.0 s, there were no significant changes in the mixing pattern and the sand volume fraction is more than 80% as in the figure above. It took a longer time to settle due to large mass per volume where the mass is more affected by the gravitational force which

acts downwards. Most of sand particles spread into water as the contour showed red-brown in color. This indicated that the concentration of sand mixed with water is larger compare to the simulation done on different impeller's shapes. Perfect mixing was established within the tank because homogeneous mixture is formed within all regions in the mixing tank.

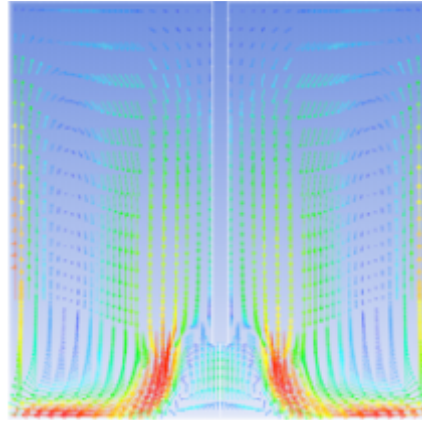


Figure 4.2.1(b): Velocity vector of sand at 250.0 s

From figure above, the sand has been suspended much higher within the mixing tank and reached the upper region of the tank. This is due to the agitation forces by impeller's blade is sufficient enough to overcome the gravity force on the sand particles inside the mixing tank. The red-brown arrows indicated the high velocity region which is near the impeller.

4.2.2 Middle Impeller

Below are the results (based on contour) of settling time for middle impeller.

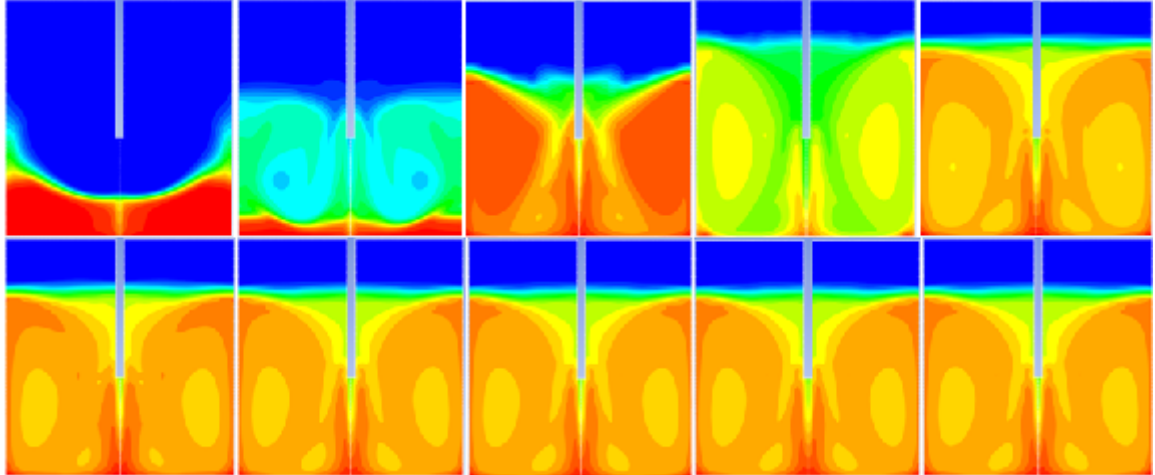


Figure 4.2.2(a): Contours of volume fraction of sand from 1.0 s to settling time of 210.0 s

This time the location of the impeller was in the middle of the mixing tank which is in water phase. It took longer time for sand particles to move because the impeller is not in touch with the particles. Upon reaching the settling time the sand equally dispersed into the water but there was a limitation for the particles to reach the top of the tank. This is because not enough force to push up the sand and also due to larger mass per volume of sand. Even though the contour look like almost the same as lower impeller, compare to the lower impeller, there is only slightly different between the settling times but lower impeller gives better mixing characteristic where homogenous mixture is formed within all regions in the mixing tank.

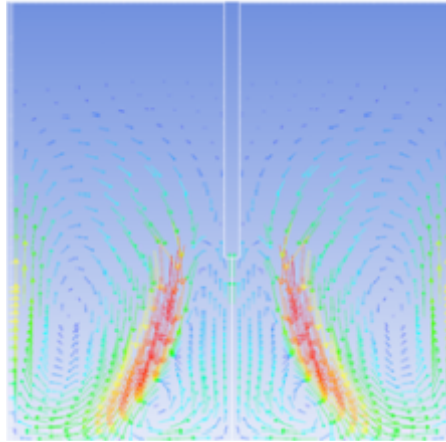


Figure 4.2.2(b): Velocity vector of sand at 210.0 s

From figure above, the sand has been suspended much higher within the mixing tank and did not reach the upper region of the tank. This is due to the agitation forces by impeller's blade is not sufficient enough to overcome the gravity force on the sand particles inside the mixing tank. The red-brown arrows indicated the high velocity region which is near the impeller.

4.2.3 Upper Impeller

Below are the results (based on contour) of settling time for upper impeller.

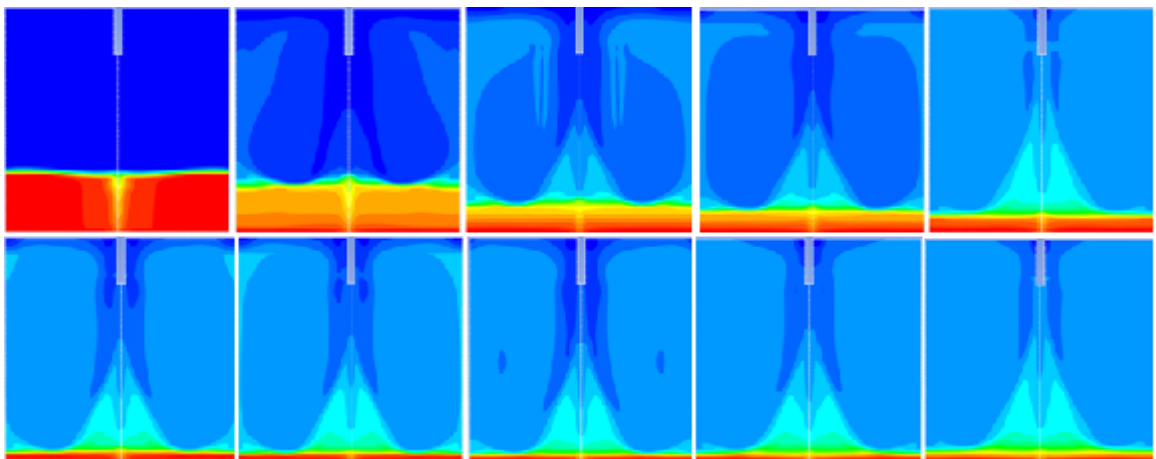


Figure 4.2.3(a): Contours of volume fraction of sand from 1.0 s to settling time of 460.0 s

The last part of simulation was to simulate the flow with impeller location at the top of the tank or the upper level within the water phase. It took the longest time for the process to reach the settling time because the location of the impeller is far away from the sand. Although the process has reached its settling time which is at 460.0 s, there was still a little amount of sand particles settled at the bottom of the tank due to not sufficient forces to overcome the gravity force on the sand particles which density is quite large. This is not efficient because other then it took the longest time the settled sand at the bottom of the tank will lead to bad mixing and high cost for maintenance to clean or remove the remaining sand after the process completed. The contour also showed that less concentration of sand particles spread into the water phase.

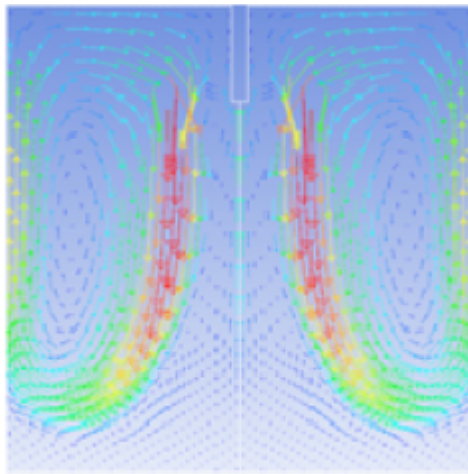


Figure 4.2.3(b): Velocity vector of sand at 460.0 s

From figure above, the sand has been suspended much higher within the mixing tank and did not reach the upper region of the tank. The velocity of sand at the bottom of the tank is the lowest in magnitude. This is due to the agitation forces by impeller's blade is not sufficient enough to overcome the gravity force on the sand particles inside the mixing tank. After settling time, there were some amount of sand settled at the bottom of the tank and formed a thin layer. The red-brown arrows indicated the high velocity region which is near the impeller.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

As conclusion, the objectives of this project to simulate the flow and mixing behavior based on the properties of the impeller and to study time taken to reach settling time and established perfect mixing were achieved.

There is only slightly difference in settling time between all type of impellers and neither of them established perfect mixing. This is due to a few reasons. First, the simulation is done for 2D problem where the domain of the model in this project is 2D axisymmetric mixing tank. We can see the effect clearly if the domain is in 3D problem. Second, the weight of impeller increased after being modified. The author created additional parts to the impeller and its weight increased. Thus, the speed of impeller will be less than before the impeller being modified. It can be noted that with an increase in the impeller rotational speed the amount of solid particles present at the bottom of the reactors has decreased (Murthy et al., 2007). Decreasing the impeller speed will affect the mixing behavior inside the tank.

For different impeller's locations, it is obviously the lower location which located in sand phase gives the better mixing characteristic than others. It's settling time also acceptable to reach perfect mixing. From the contour at settling time of 250.0s, homogeneous mixture established where the sand volume fraction is spread homogeneously in the mixing tank where the contour of sand consist of majorly (approximately more than 80%) of a single range of volume fraction. It took a longer time to settle due to large mass per volume where the mass is more affected by the gravitational force which acts downwards. Lower impeller gives better mixing characteristic where homogenous mixture is formed within all regions in the mixing tank.

The different in contours was because of the concentration of sand that spread into water phase. Most contours for different types of impeller showed the blue-green contour to represent the water-sand mixture because only small amount of sand particle mix with water and the rest is trapped below the impeller and settled down at the bottom due to high density. The contours for impeller's location showed red-brown contour due to high amount of sand particle mix with water. In this case, only density of sand affects the mixing behavior inside the tank. As you can see that the middle and upper location of impeller showed no perfect mixing established at their settling time.

As recommendation, this project can be done in 3-dimensional (3D) problem in order to get better and accurate results. Based from the result to analyze perfect mixing, the number of scale of the level should be reduce from 20 to 5 so that it would be easier to observe the perfect mixing condition. Further study should be conducted especially in meshing work to produce a fine mesh file for each problem. This step will be a crucial step because it will affect during simulation work. Since the model is in 3D, longer computational time is required to complete the simulation. Other aspect is to use other type of mixture instead of using water-sand mixture. More viscous liquid will be used so that the effect of impeller properties like shape, blade width, and number of blade can be investigated. In this work, speed of the impeller is constant for each problem. In future work, I suggest to perform the simulation with different impeller speed.

REFERENCES

- Aubin, J., & Xuereb, C. (2006). Design of multiple impeller stirred tanks for the mixing of highly viscous fluids using CFD. *Chemical Engineering Science*, 61(9), 2913-2920.
- Buwa, V., Dewan, A., Nassar, A. F., & Durst, F. (2006). Fluid dynamics and mixing of single-phase flow in a stirred vessel with a grid disc impeller: Experimental and numerical investigations. *Chemical Engineering Science*, 61(9), 2815-2822.
- Cornelissen, J. T., Taghipour, F., Escudié, R., Ellis, N., & Grace, J. R. (2007). CFD modelling of a liquid–solid fluidized bed. *Chemical Engineering Science*, 62(22), 6334-6348.
- Jahoda, M., Tomášková, L., & Moštko, M. (2009). CFD prediction of liquid homogenisation in a gas–liquid stirred tank. *Chemical Engineering Research and Design*, 87(4), 460-467.
- Kumaresan, T., & Joshi, J. B. (2006). Effect of impeller design on the flow pattern and mixing in stirred tanks. *Chemical Engineering Journal*, 115(3), 173-193.
- Murthy, B. N., Deshmukh, N. A., Patwardhan, A. W., & Joshi, J. B. (2007). Hollow self-inducing impellers: Flow visualization and CFD simulation. *Chemical Engineering Science*, 62(14), 3839-3848.
- Murthy, B. N., Ghadge, R. S., & Joshi, J. B. (2007). CFD simulations of gas–liquid–solid stirred reactor: Prediction of critical impeller speed for solid suspension. *Chemical Engineering Science*, 62(24), 7184-7195.
- Nasrallah, N., Legrand, J., Bensmaili, A., & Nouri, L. (2008). Effect of impeller type on the mixing in torus reactors. *Chemical Engineering and Processing: Process Intensification*, 47(12), 2175-2183.

- Nienow, A. W. (1997). On impeller circulation and mixing effectiveness in the turbulent flow regime. *Chemical Engineering Science*, 52(15), 2557-2565.
- Pedrosa, S. M. C. P., & Nunhez, J. R. (2000). The behavior of stirred vessels with anchor type impellers. *Computers & Chemical Engineering*, 24(2-7), 1745-1751.
- Rahimi, M., Senior, P. R., & Mann, R. (2000). Visual 3-D modelling of stirred vessel mixing for an inclined-blade impeller. *Chemical Engineering Research and Design*, 78(3), 348-353.
- Shewale, S. D., & Pandit, A. B. (2006). Studies in multiple impeller agitated gas–liquid contactors. *Chemical Engineering Science*, 61(2), 489-504.
- Szalai, E. S., Arratia, P., Johnson, K., & Muzzio, F. J. (2004). Mixing analysis in a tank stirred with ekato intermig® impellers. *Chemical Engineering Science*, 59(18), 3793-3805.
- Fluent Inc., (2006). Fluent Software Tutorial Guide, *Tutorial 20. Using the Eulerian Multiphase Model for Granular Flow*.
- Fluent Inc., (2006). Software User Guide.
- Fluent Inc., (2006). Gambit Software Tutorial Guide.
- Bakker A, 2006, <http://www.bakker.org>, 18 February 2010.
- Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University, (2009). *FLUENT – Unsteady Flow Past a Cylinder*,
<https://confluence.cornell.edu/display/SIMULATION/FLUENT+-+Unsteady+Flow+Past+a+Cylinder+-+Step+5> , 10 March 2010.
- <http://www.wikipedia.com>, 17 February 2010.
- http://www.chmltech.com/cfd/grid_generation.pdf, 18 February 2010.

APPENDICES

The value of the time averaged impeller velocity components and turbulence quantities are based on experimental measurement. The variation of these values may be expressed as a function of radius, and imposed as polynomials according to:

$$\text{Variable} = A_1 + A_2r + A_3r^2 + A_4r^3 + \dots$$

This macro is usually used to specify a profile condition on a boundary face zone; it is used in fix.c to specify the condition in a fluid cell zone. The arguments of the macro have been changed accordingly.

Table 1: Impeller profile specification

Variable	A_1	A_2	A_3	A_4	A_5	A_6
u velocity	-7.1357e-2	54.304	-3.1345e+3	4.5578e+4	-1.966e+5	—
v velocity	3.1131e-2	-10.313	9.5558e+2	-2.0051e+4	1.186e+5	—
kinetic energy	2.2723e-2	6.7989	-424.18	9.4615e+3	-7.725e+4	1.8410e+5
dissipation	-6.5819e-2	88.845	-5.3731e+3	1.1643e+5	-9.120e+5	1.9537e+6

fix.c:

```
#include "udf.h"
#include "sg.h"

#define FLUID_ID 1
#define ua1 -7.1357e-2
#define ua2 54.304
#define ua3 -3.1345e3
#define ua4 4.5578e4
#define ua5 -1.9664e5

#define va1 3.1131e-2
#define va2 -10.313
#define va3 9.5558e2
#define va4 -2.0051e4
#define va5 1.1856e5

#define ka1 2.2723e-2
#define ka2 6.7989
#define ka3 -424.18
```

```

#define ka4 9.4615e3
#define ka5 -7.7251e4
#define ka6 1.8410e5

#define da1 -6.5819e-2
#define da2 88.845
#define da3 -5.3731e3
#define da4 1.1643e5
#define da5 -9.1202e5
#define da6 1.9567e6

DEFINE_PROFILE(fixed_u, thread, np)
{
    cell_t c;
    real x[ND_ND];
    real r;

    begin_c_loop (c,thread)
    {
        /* centroid is defined to specify position dependent profiles*/
        C_CENTROID(x,c,thread);
        r = x[1];
        F_PROFILE(c,thread,np) =
            ua1 +(ua2*r)+(ua3*r*r)+(ua4*r*r*r)+(ua5*r*r*r*r);
    }
    end_c_loop (c,thread)
}

DEFINE_PROFILE(fixed_v, thread, np)
{
    cell_t c;
    real x[ND_ND];
    real r;

    begin_c_loop (c,thread)
    {
        /* centroid is defined to specify position dependent profiles*/
        C_CENTROID(x,c,thread);
        r = x[1];
        F_PROFILE(c,thread,np) =
            va1 +(va2*r)+(va3*r*r)+(va4*r*r*r)+(va5*r*r*r*r);
    }
    end_c_loop (c,thread)
}

```

```

DEFINE_PROFILE(fixed_ke, thread, np)
{
    cell_t c;
    real x[ND_ND];
    real r;

    begin_c_loop (c,thread)
    {
        /* centroid is defined to specify position dependent profiles*/
        C_CENTROID(x,c,thread);
        r = x[1];
        F_PROFILE(c,thread,np) =
            ka1 + (ka2*r) + (ka3*r*r) + (ka4*r*r*r) + (ka5*r*r*r*r) + (ka6*r*r*r*r*r);

    }
    end_c_loop (c,thread)
}

DEFINE_PROFILE(fixed_diss, thread, np)
{
    cell_t c;
    real x[ND_ND];
    real r;

    begin_c_loop (c,thread)
    {
        /* centroid is defined to specify position dependent profiles*/
        C_CENTROID(x,c,thread);
        r = x[1];
        F_PROFILE(c,thread,np) =
            da1 + (da2*r) + (da3*r*r) + (da4*r*r*r) + (da5*r*r*r*r) + (da6*r*r*r*r*r);

    }
    end_c_loop (c,thread)
}

```

Month	Feb 2010				Mar 2010				Apr 2010				May 2010			
Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
FYP briefing and seminar																
Proposal preparation and submission																
Mixing and impeller search																
Study on mixing behavior																
Study on the effect of impeller shape and speed on mixing behavior																
Perform Fluent Tutorial to simulate the flow in the mixing tank																
Mid Semester Break																
Submission of Progress Report																
Seminar																
Study on meshing the impeller using GAMBIT																
Perform simulation-determine time step size																
Interim Report																
Oral Presentation																

Figure 1: Gantt chart for Final Year Project 1

Month	Aug 2010				Sep 2010				Oct 2010				Nov 2010			
Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Project Work Continue (create mesh files, run simulation)																
Submission of Progress Report 1																
Project Work Continue (add buffer and run simulation)																
Submission of Progress Report 2																
Seminar (compulsory)																
Project Work Continue (collect all data and analyze data)																
Poster Exhibition																
Submission of Dissertation (soft bound)																
Oral Presentation	Week 18 - 19															
Submission of Project Dissertation (Hard Bound)	Before 17 December 2010															

Figure 2: Gantt chart for Final Year Project 2